Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-3010: Fuselage Sizing Trade-Off using Categorical Variables

HS-3010: Fuselage Sizing Trade-Off using Categorical Variables

The purpose of this tutorial is to investigate the relative effect of the variable on the identified output responses. Furthermore, this tutorial will demonstrate how to create a Fit in order to investigate combinations of variables that were not explicitly simulated.

Three continuous variables and three categorical variables are used in this tutorial. The frames can take five possible sections, and the stringers can each take from four available sections.

Continuous variables

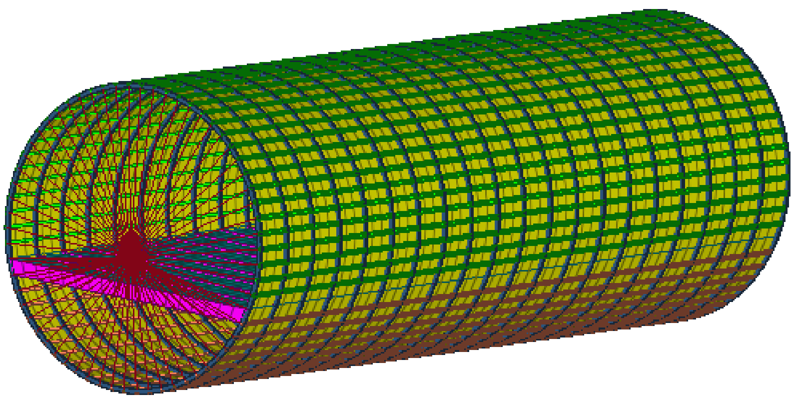
- Thickness of floor
- Thickness of floor beams
- Skin thickness

Category variables

- Cross sections of the frames
- Stringers above the floor
- Stringers below the floor

This tutorial uses three load cases:

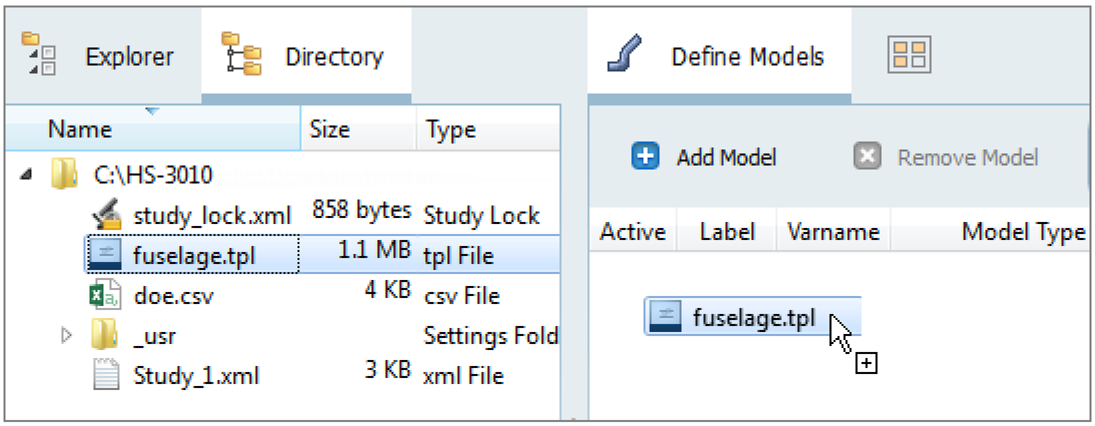
- Free-free normal modes case
- Simple bending case
- Simple torsional case



The files used in this tutorial can be found in `<hst.zip>/HS-3010/`. Copy the files from this directory to your working directory.

Step 1: Perform the Study Setup

1. Start HyperStudy.
2. To start a new study, click ***File*** > ***New*** from the menu bar, or click on the toolbar.
3. In the **HyperStudy – Add** dialog, enter a study name, select a location for the study, and click ***OK***.
4. Go to the **Define models** step.
5. Add a Parameterized File model.
   a. From the **Directory**, drag-and-drop the `fuselage.tpl` file into the work area.



   b. In the **Solver input file** column, enter `fuselage.fem`. This is the name of the solver input file HyperStudy writes during any evaluation.
   c. In the **Solver execution script** column, select ***OptiStruct (os)***.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	☑	Model 1	m_1	{} Parameterized File	C:/.../HS-3010/fuselage.tpl	fuselage.fem	OptiStruct (os)	${file}

6. Click ***Import Variables***. Six input variables are imported from the `fuselage.tpl` resource file.
7. Go to the **Define Input Variables** step.
8. Review the input variable's lower and upper bounds ranges.
9. Go to the **Specifications** step.

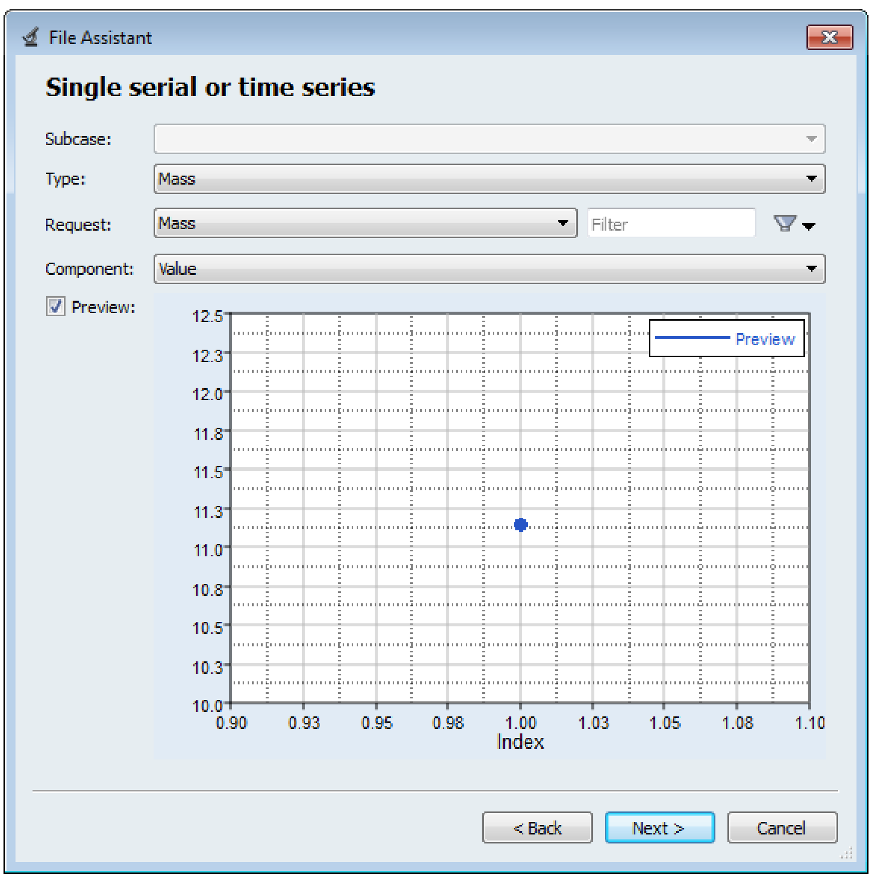
Step 2: Perform the Nominal Run

1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.
3. Go to the **Evaluate** step.

4. Click ***Evaluate Tasks***.
5. Go to the **Define Output Responses** step.

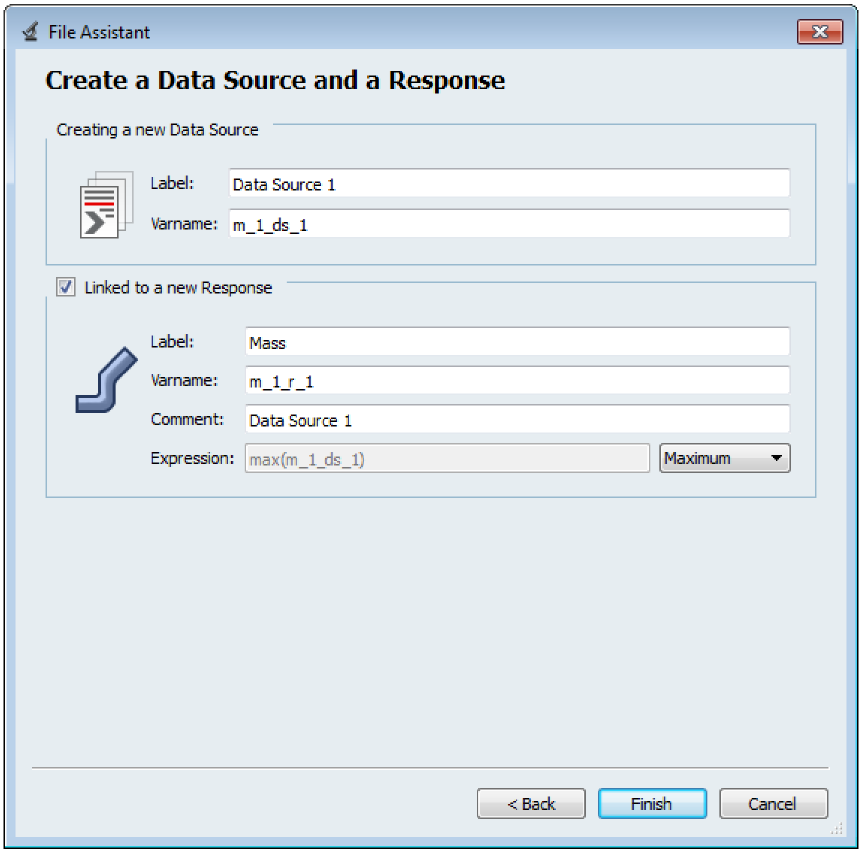
Step 3: Create and Define Output Responses

1. Create the Mass output response.
    a. From the **Directory**, drag-and-drop the `fuselage.out` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.
    b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (hgosfreq.exe)*** and click ***Next***.
    c. Select **Single item in a time series**, then click ***Next***.
    d. Define the following options, and then click ***Next***.
        - Set **Type** to ***Mass***.
        - Set **Request** to ***Mass***.
        - Set **Component** to ***Value***.



    e. Label the output response Mass.

f. Set **Expression** to ***Maximum***.



g. Click ***Finish***. The Mass output response is added to the work area.

2. Create two more output responses by repeating step 1, except change the type, request, and component assigned to each output response to the following.

   Because this is a free-free analysis, Freq1 will be the seventh frequency in the list due to the six rigid body modes (all near zero). Freq2 will be the eighth frequency in the list.

Output Response	Type	Request	Component
Freq1	Frequency	Mode 7	Value
Freq2	Frequency	Mode 8	Value

3. Create the Bending displacement output response, which will have a magnitude of node 8196 (loading point).

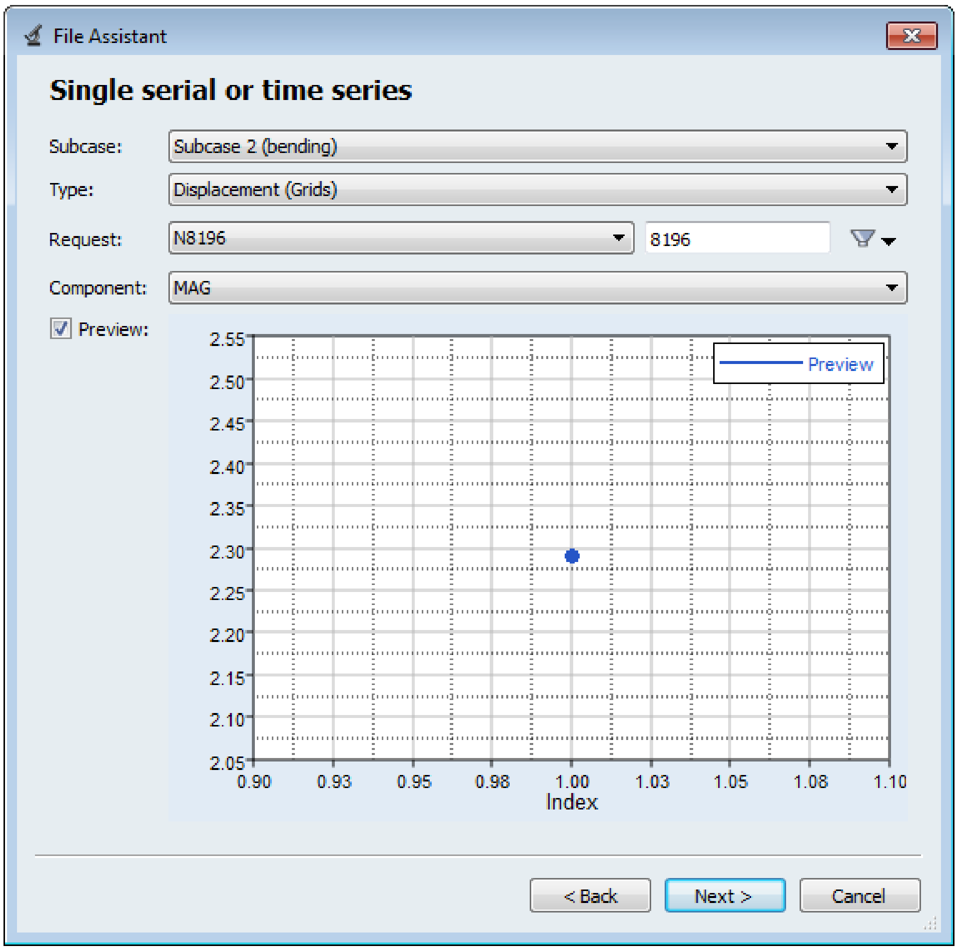
   a. From the **Directory**, drag-and-drop the `fuselage.h3d` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.

   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (Hyper3D Reader)*** and click ***Next***.

c. Select **Single item in a time series**, then click ***Next***.

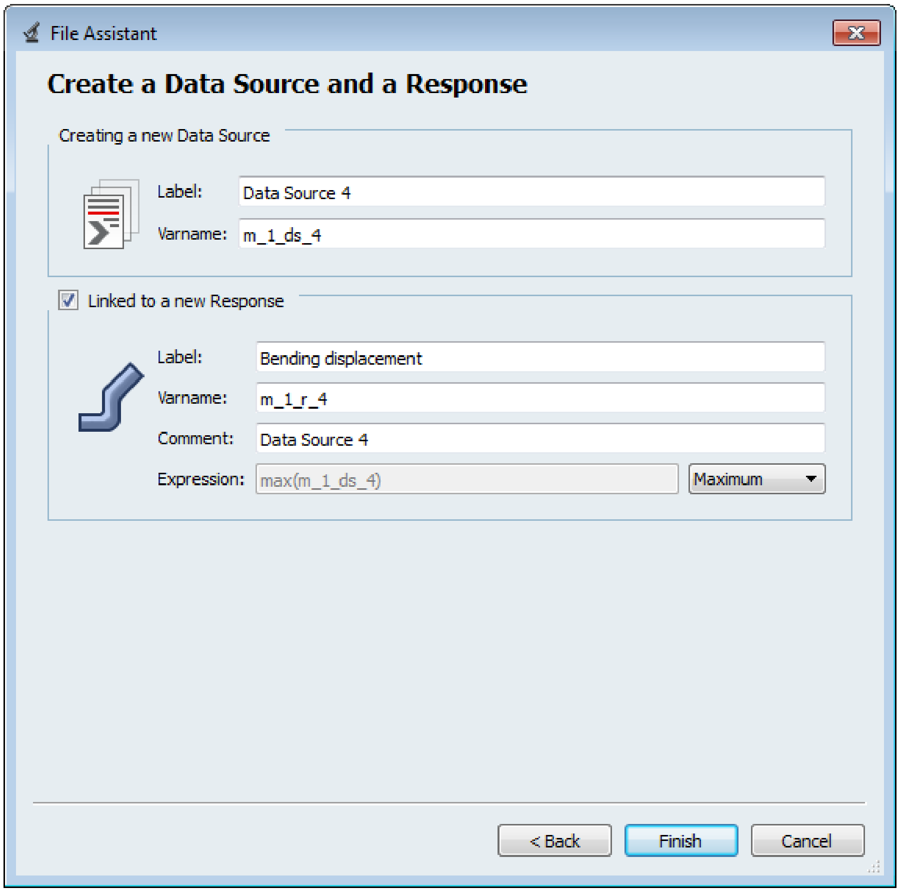
d. Define the following options, and then click ***Next***.

- Set **Subcase** to ***Subcase 2 (bending)***.
- Set **Type** to ***Displacement(Grids)***.
- For **Request**, apply a filter of 8196. Press ***Enter*** to accept the value entered in the **Filter** field.
- Set **Component** to ***Mag***.



e. Label the output response Bending displacement.

f. Set **Expression** to ***Maximum***.



g. Click ***Finish***. The Bending displacement output response is added to the work area.

4. Create the Torsional rotation output response, which will have a z-direction of node 8196 (loading point).

   a. From the **Directory**, drag-and-drop the `fuselage.h3d` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.

   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (Hyper3D Reader)*** and click ***Next***.

   c. Select **Single item in a time series**, then click ***Next***.

   d. Define the following options, and then click ***Next***.

      - Set **Subcase** to ***Subcase 3 (torsion***).
      - Set **Type** to ***Rotation (Grids)***.
      - For **Request**, apply a filter of 8196. Press ***Enter*** to accept the value entered in the **Filter** field.
      - Set **Component** to ***Z***.

   e. Label the response Torsional rotation.

   f. Set **Expression** to ***Maximum***.

   g. Click ***Finish***. The Torsional rotation output response is added to the work area.

h. In the **Expression** field for Torsional rotation, edit the expression to be `max(m_1_ds_5)*360/3.14`.

This expression converts the rotation from radians to degrees.

5. Click ***Evaluate Expressions*** to extract output response values.

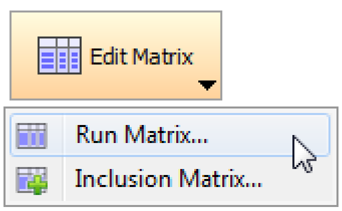
	Active	Label	Varname	Expression	Value	Comment
1	☑	Mass	m_1_r_1	max(m_1_ds_1) ...	Not Extracted	Data Source 1 ...
2	☑	Freq1	m_1_r_2	max(m_1_ds_2) ...	Not Extracted	Data Source 2 ...
3	☑	Freq2	m_1_r_3	max(m_1_ds_3) ...	Not Extracted	Data Source 3 ...
4	☑	Bending displacement	m_1_r_4	max(m_1_ds_4) ...	Not Extracted	Data Source 4 ...
5	☑	Torsional rotation	m_1_r_5	max(m_1_ds_5)*360/3.14 ...	Not Extracted	Data Source 5 ...

6. Click ***OK***. This complete the study setup.

Step 4: Run a Doe

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Doe*** and click ***OK***.
3. Go to the **Specifications** step.
4. In the work area, set the **Mode** to ***None***.
5. Click ***Apply***.
6. Import run data using the Run Matrix.

a. Click ***Edit Matrix*** > ***Run Matrix*** from the top, right corner of the work area.



b. In the **Edit Data Summary** dialog, remove any existing run data.

c. Click ***Import Values***.

d. In the **Import Values** dialog, select ***Plain Text*** and click ***Next***.

e. In the **Source File** field, navigate to the `doe.csv` file and click ***Next***.

f. Click ***Finish***.

g. Review the imported run data and click ***Apply***.

h. Click ***OK***.

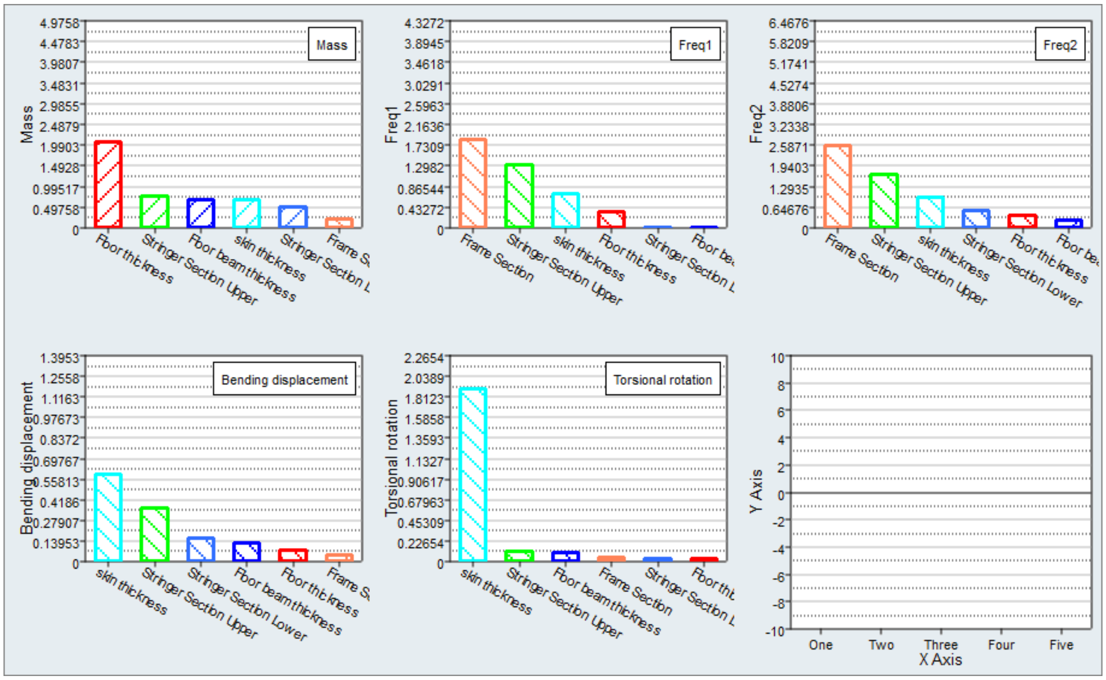
	Frame Section	Stringer Section Upper	Stringer Section Lower	Floor thickness	Floor beam thickness	skin thickness
1	c-1.00x2.25b0.125	I-0.50x1.00b0.125	hat-1.00x0.75b0.125	0.4000000	0.3000000	0.0400000
2	z-1.00x2.00b0.100	hat-1.00x0.75b0.125	I-0.50x0.50b0.100	0.6000000	0.3000000	0.0600000
3	z-1.00x1.75b0.125	I-0.50x0.50b0.100	I-0.50x0.50b0.100	0.4000000	0.5000000	0.0400000
4	z-1.00x2.00b0.100	I-0.50x0.50b0.100	hat-1.00x1.00b0.100	0.6000000	0.5000000	0.0400000
5	c-1.00x2.00b0.100	hat-1.00x0.75b0.125	hat-1.00x1.00b0.100	0.6000000	0.5000000	0.0600000
6	c-1.00x2.50b0.100	hat-1.00x1.00b0.100	I-0.50x1.00b0.125	0.6000000	0.5000000	0.0600000
7	z-1.00x2.00b0.100	hat-1.00x1.00b0.100	I-0.50x0.50b0.100	0.6000000	0.3000000	0.0400000
8	z-1.00x1.75b0.125	I-0.50x0.50b0.100	hat-1.00x1.00b0.100	0.4000000	0.3000000	0.0400000
9	z-1.00x1.75b0.125	I-0.50x0.50b0.100	hat-1.00x0.75b0.125	0.6000000	0.3000000	0.0600000
10	z-1.00x1.75b0.125	hat-1.00x1.00b0.100	hat-1.00x1.00b0.100	0.4000000	0.5000000	0.0600000
11	c-1.00x2.50b0.100	hat-1.00x0.75b0.125	hat-1.00x1.00b0.100	0.6000000	0.3000000	0.0400000
12	c-1.00x2.50b0.100	I-0.50x1.00b0.125	hat-1.00x0.75b0.125	0.4000000	0.5000000	0.0600000
13	z-1.00x1.75b0.125	hat-1.00x0.75b0.125	I-0.50x1.00b0.125	0.4000000	0.3000000	0.0400000
14	c-1.00x2.50b0.100	hat-1.00x1.00b0.100	hat-1.00x1.00b0.100	0.6000000	0.3000000	0.0600000
15	z-1.00x1.75b0.125	hat-1.00x0.75b0.125	I-0.50x1.00b0.125	0.4000000	0.3000000	0.0600000
16	c-1.00x2.00b0.100	hat-1.00x0.75b0.125	I-0.50x0.50b0.100	0.6000000	0.3000000	0.0400000
17	c-1.00x2.25b0.125	hat-1.00x1.00b0.100	I-0.50x0.50b0.100	0.4000000	0.3000000	0.0400000

7. Go to the **Evaluate** step.
8. Click ***Evaluate Tasks*** to run the Doe.
9. Go to the **Post-Processing** step, and click the ***Pareto Plot*** tab. Enable **multi-plot** and select all of the output responses in the **Channel** selector. If not all of the input variables are plotted, you may need to alter the number of displayed input variables from the options menu.

   The relative effect of a input variable can vary from output response to output response. The most influential input variables when analyzing frequency output responses are Frame Section and Stringer Section Upper. In contrast, the most influential input variables when analyzing the two stiffness conditions are Skin thickness and Stringer Section Upper.

   Some input variables can have no effect on output responses. Floor beam thickness has minimal effect on any of the output responses, which indicates that you may want to consider removing this input variable from the analysis.

   In a Pareto plot, the effect of input variables on output responses does not measure sensitivity but rather absolute change. Floor thickness has a major effect on Volume. This effect is not a derivative, but a measure of the possible increase over the range of the input variables (the range is the difference between the upper and lower bounds). The floor has a large area and the thickness has very large bounds (+/-0.1 inches), therefore it can make a dramatic impact on Volume as the input variables move through the available space.



Step 5: Run a Least Sqaures Regression Fit

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Fit*** and click ***OK***.
3. Go to the **Select Matrices** step.
4. Click ***Add Matrix***.
5. In the **Add - HyperStudy** dialog, add one matrix.
6. Define the matrix.
   a. Set **Type** to ***Input***.
   b. Set **Matrix Source** to ***Doe1 (doe_1)***.

	Active	Label	Varname	Type	Matrix Source	Matrix Origin	Status
1	☑	FitMatrix 1	fitmatrix_1	Input	Doe 1 (doe_1)	DoeDoe 1	Import Pending

7. Click ***Import Matrix***.
8. Go to the **Specifications** step.
9. In the work area, set the **Mode** to ***Least Squares Regression***.
10. Click ***Apply***.
11. Go to the **Evaluate** step.
12. Click ***Evaluate Tasks*** to evaluate the designs.

13. Go to the **Post processing** step.
14. Click the ***Diagnostics*** tab to assess the accuracy of the Fit. Select the Mass output response in the **Channel** selector.

    The R-Square and R-Squared Adjusted values for Mass are 1.00, which indicates the model perfectly predicted the known values.

	Criterion	Input Matrix	Cross-Validation Matrix	Validation Matrix
1	R-Square	1.0000000	1.0000000	N/A
2	R-Square Adjusted	1.0000000	N/A	N/A
3	Multiple R	1.0000000	1.0000000	N/A
4	Relative Average Absolute Error	1.15e-05	1.60e-05	N/A
5	Maximum Absolute Error	4.04e-05	5.11e-05	N/A
6	Root Mean Square Error	1.70e-05	2.41e-05	N/A
7	Number of Samples	50	50	0

15. Review the diagnostics for the remaining output responses. Notice the R-Squared values are still high, which indicates a high quality fitting of the data.
16. Click the ***ANOVA*** tab and review the **Mean Squares Percent** column to see the relative importance of input variables. The results should be similar to the results noted in the **Pareto Chart** tab of the Doe.
17. Click the ***Trade-Off*** tab to perform "what if" scenarios. In the **Inputs** pane, modify the values of input variables to see their effect on the output response approximations in the **Output** pane.

Last modified: v2017.2 (12.1156684)